

Simulations of the Nozzle Panel Forming Process Using ABAQUS

by
Jim Mahoney (AS/FM&T)

**AlliedSignal Inc.,
Federal Manufacturing & Technologies
is operated for the U.S. Department
of Energy under contract number
DE-ACO4-76DP00613**

Virtual Manufacturing

Annual Partners Meeting • March 27 - 28, 1996



Simulation using ABAQUS

Overview

- Recap of overall simulation process
- Forming simulations and trials
- Resources utilized for process
- Overall benefit/expense for this type application

Simulation using ABAQUS

Simulation Process

- ⌚ Cad translation to FEA complete (08/95)
- ⌚ Initial meshes of blank, holders, and Dies (10/95)
- ⌚ Explicit dynamic preliminary forming (12/96)
- ⌚ Iterations on loadings and timing (02-3/96)
- ⌚ Elastic rebound for net shape (03/96)

Simulation using ABAQUS

Simulation Input

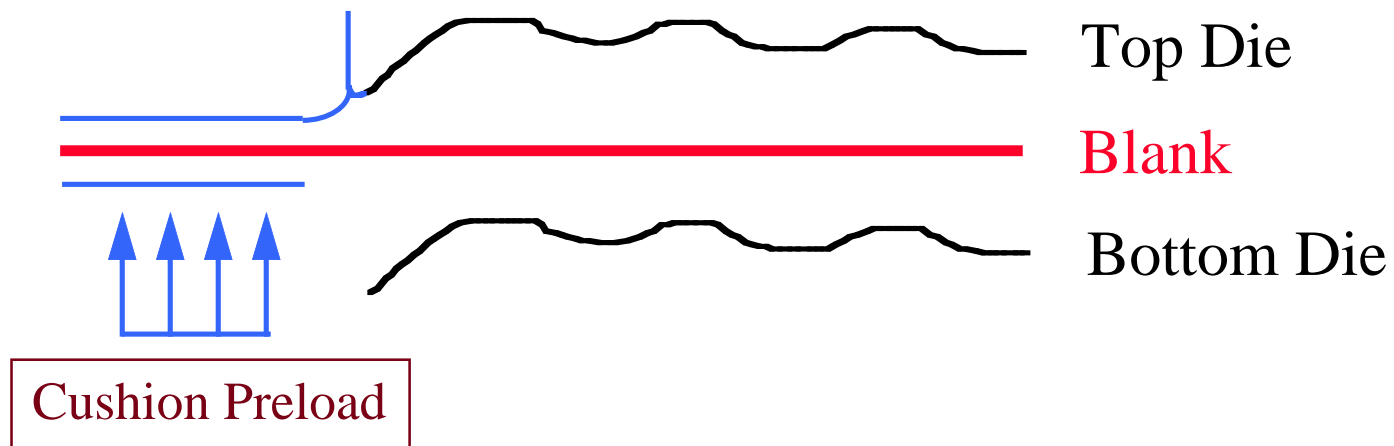
- Blank geometry
- Die geometries
- Blank material/condition
- Press forming rates
- Blank cushion pressure

Simulation Output

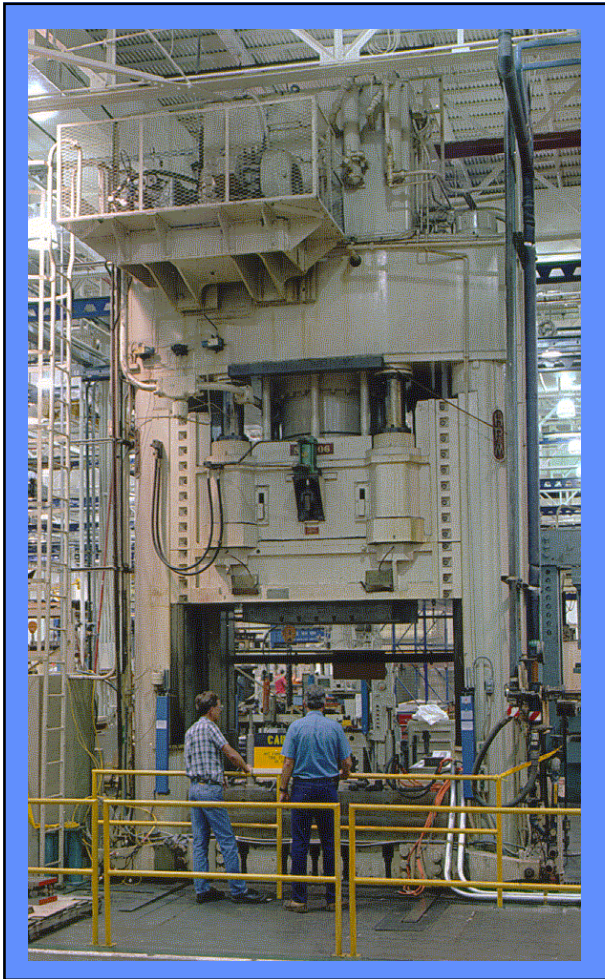
- Forming loads
- Final part shape
- Material Damage

Simulation using ABAQUS

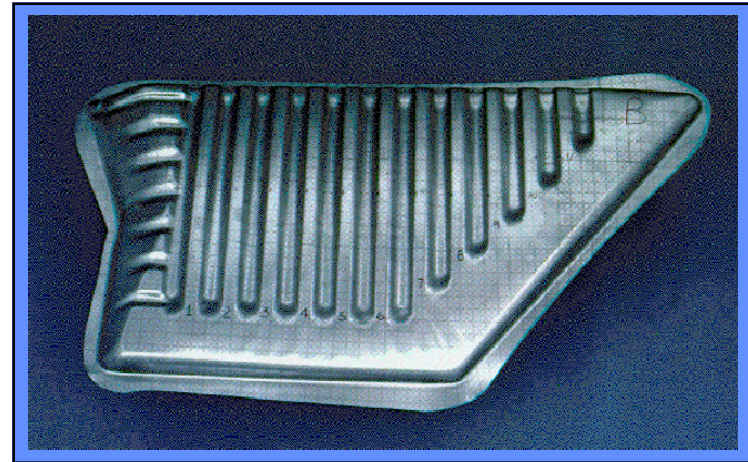
Sheet Metal Nozzle Panel Forming



Simulation using ABAQUS



Sheet Metal Nozzle Panel
Forming

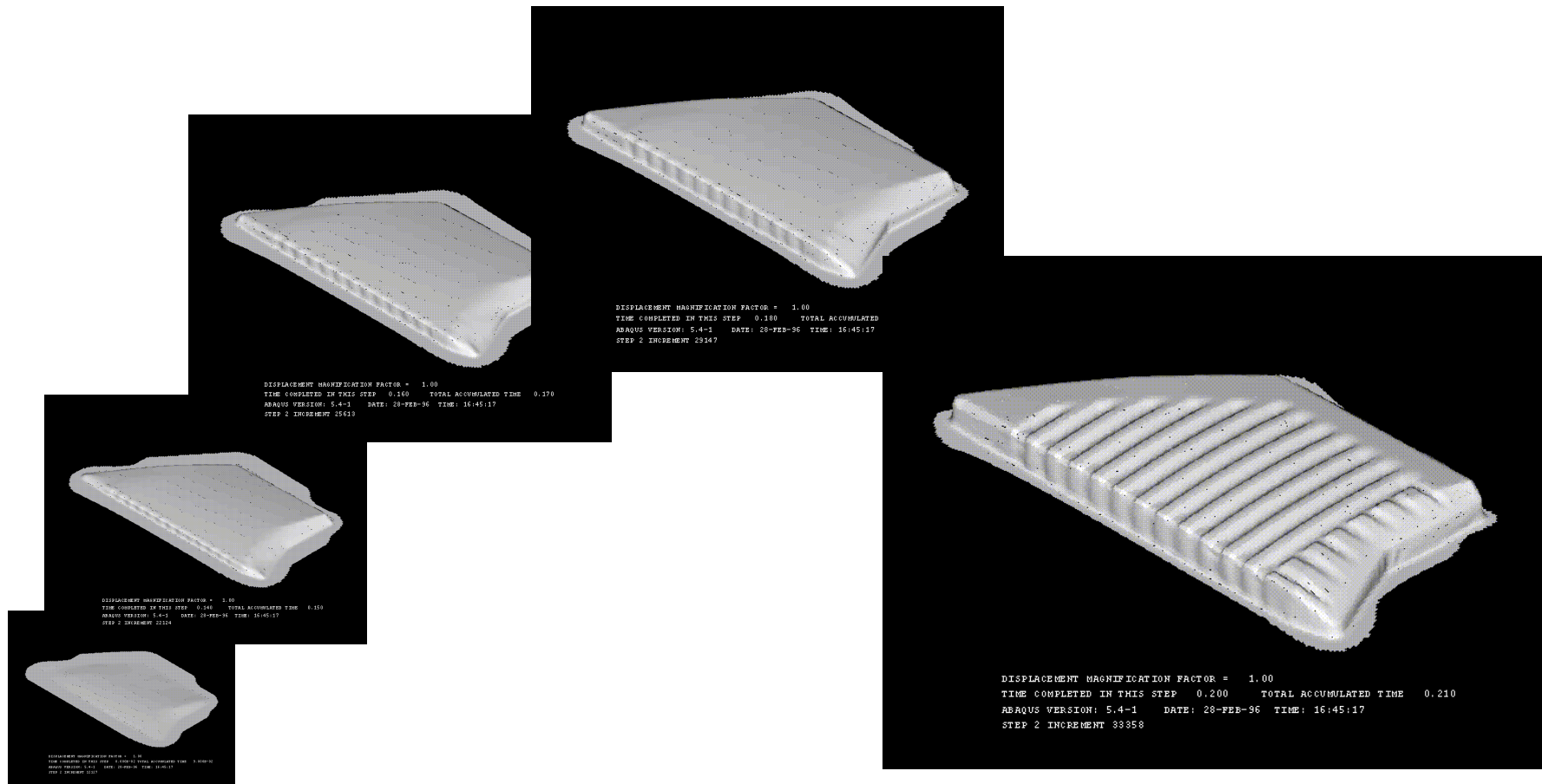


Simulation using ABAQUS

Simulation Trials Performed

- ✿ 20-30 runs processed
- ✿ 0.032" Inconel 718 blank
- ✿ Varying cushion preload
- ✿ 0.05 blank/die friction
- ✿ time independent modeling

Simulation using ABAQUS



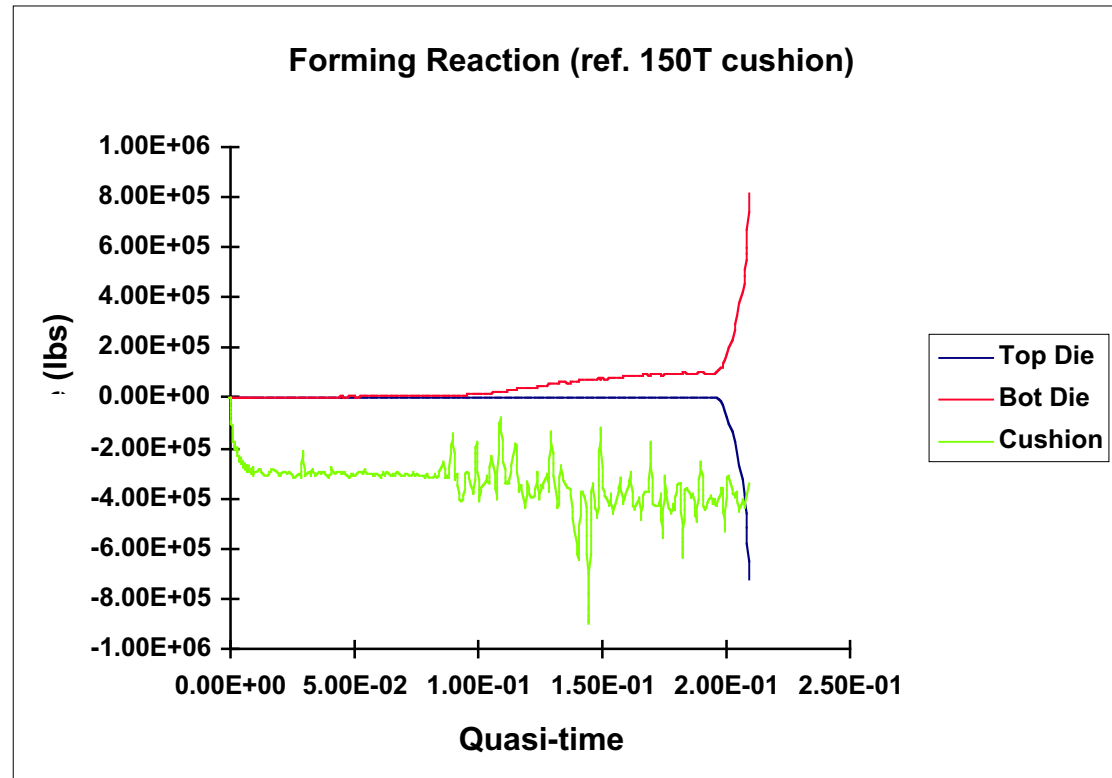
Ref: clamp21 run

Virtual Manufacturing

Annual Partners Meeting • March 27 - 28, 1996



Simulation using ABAQUS



Ref: clamp23 run

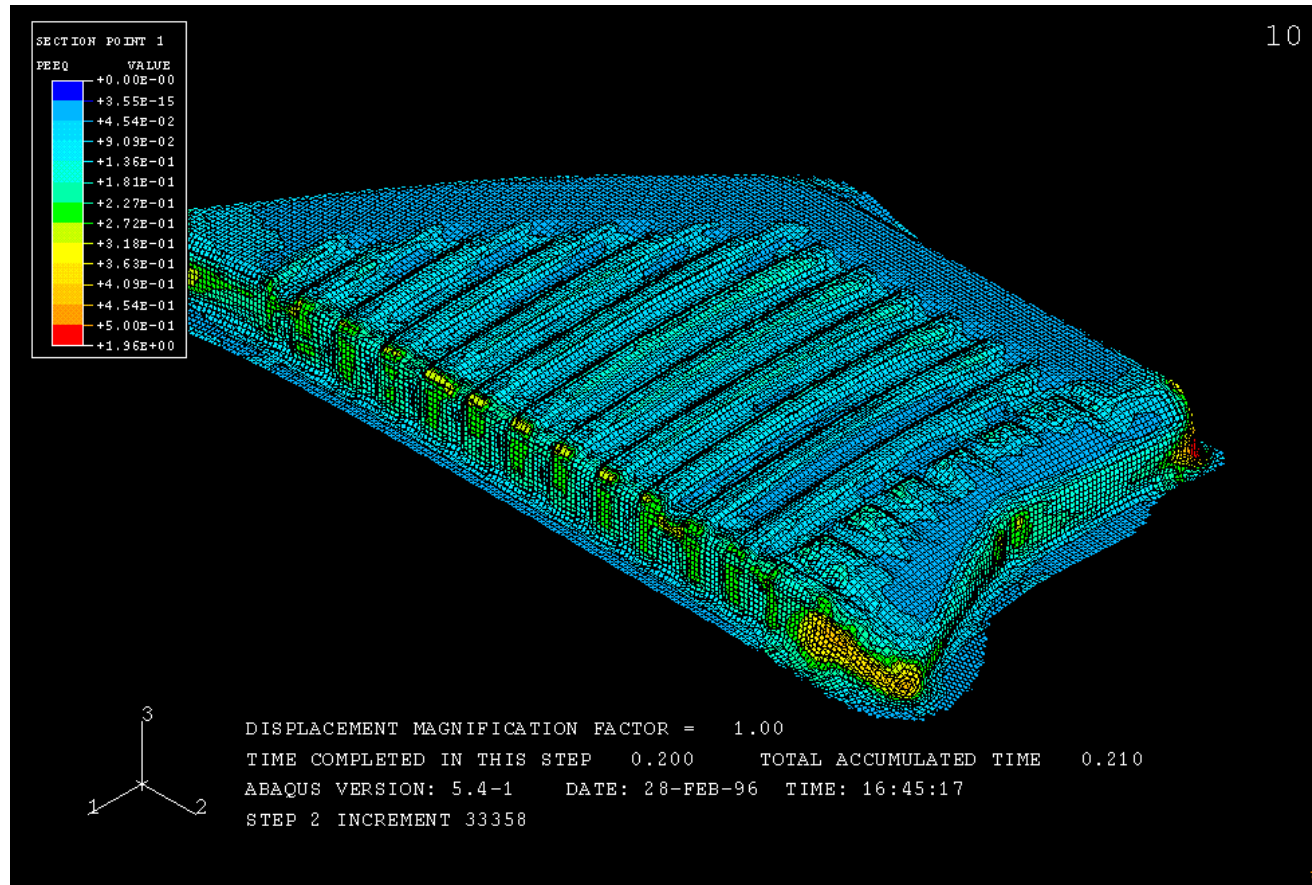
Virtual Manufacturing

Annual Partners Meeting • March 27 - 28, 1996



Simulation using ABAQUS

Effective Plastic Strains @ final form - 50T cushion



Ref: clamp21 run

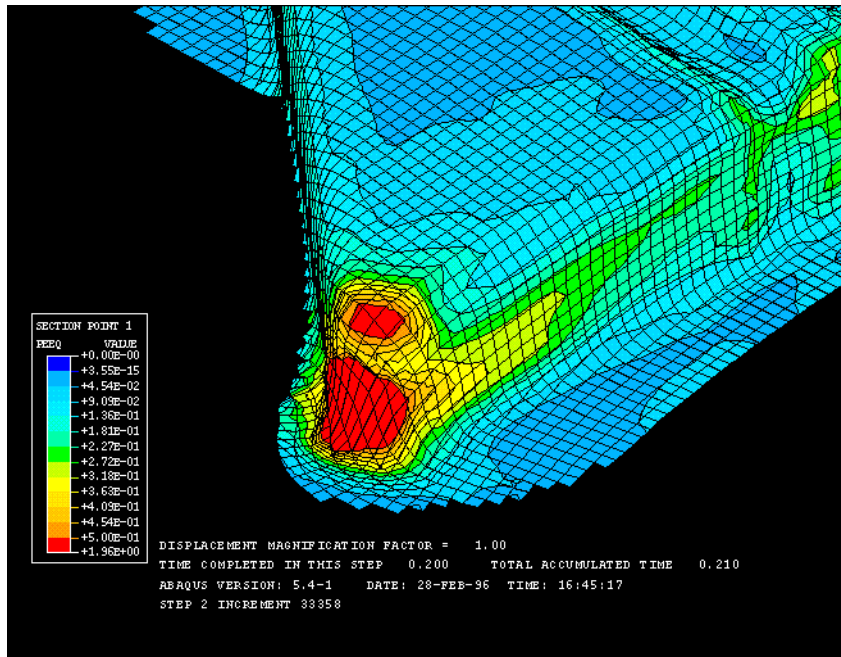
Virtual Manufacturing

Annual Partners Meeting • March 27 - 28, 1996



Simulation using ABAQUS

Effective Strain in corner (max)



Cushion Preload	Strain
15T	0.521
50T	0.889
150T	0.979

Ref: clamp21 run

Simulation using ABAQUS

Final Shape - Explicit w/Implicit rebound

FEA Simulation



Actual Part

Virtual Manufacturing

Annual Partners Meeting • March 27 - 28, 1996



Simulation using ABAQUS

Simulation Times

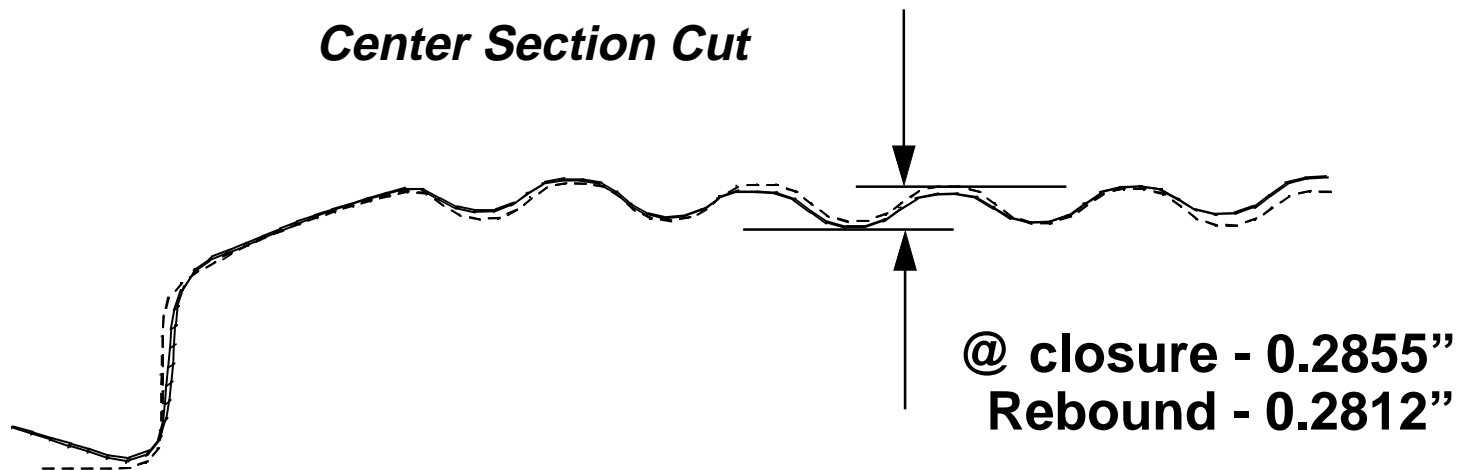
	Cray J-90	Sun Sparc 20
Cushion Preload*	2:45	5:00
Forming ~ 2"	6:00	12:00
Rebound (Implicit)	2:00	4:00

Simulation times varies from 0.5 to 59 hrs,
depending on dynamic response of system

*cpu times in hrs

Simulation using ABAQUS

Elastic Rebound of bead area

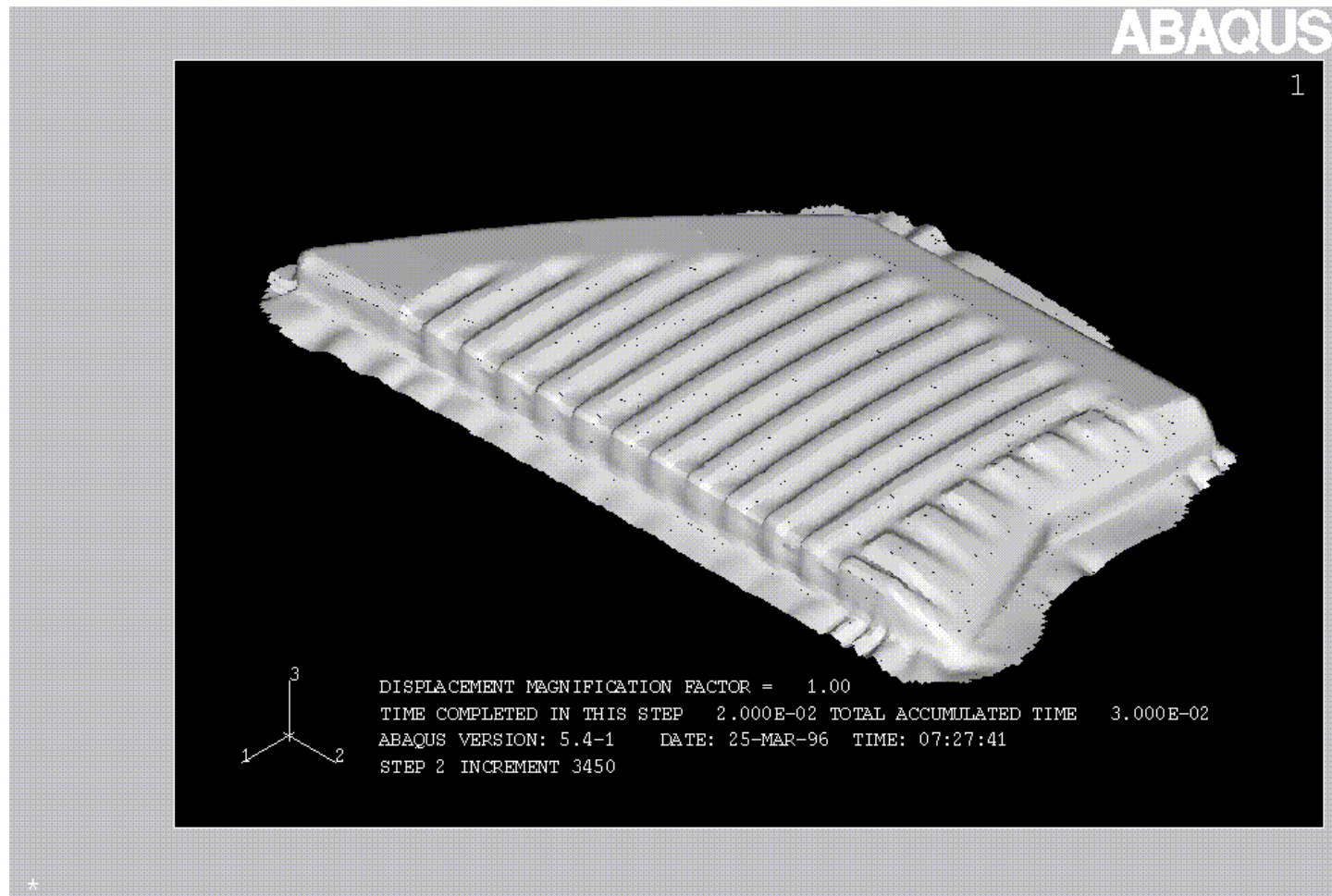


Ref: clamp21 run

Simulation using ABAQUS

Wrinkling

Cushion load - 3000 lbs



Virtual Manufacturing

Ref: clamp21d run



Simulation using ABAQUS

Simulation Results

Comparison between test and analysis

- ⚙ Strains in nose areas highest -
unknown failure point
- ⚙ Strains in transverse bead high - but
not alarming
- ⚙ Lower cushion pressures caused
slight blank wrinkling - poor correlation
to test data point
- ⚙ Elastic rebound in part matches
twisted shape found in testing
- ⚙ Little elastic rebound in bead area

Simulation using ABAQUS

Simulation flowtime

For this panel.....

Preprocessing CAD data	8 weeks
Preparing meshes	6 weeks
Solutions	20 weeks
total - 34 weeks	

Issues -

- CAD processing almost eliminated in solid case
- Mesh process 2X too long because of CAD data
- Solutions could be compressed under full-time scenario

Optimistic 3 month flow

Simulation using ABAQUS

Is this computer simulation technology ready for design-for-manufacturing on these type parts?

Pro's -

- ☺ Understanding problem areas
- ☺ Elastic rebound simple
- ☺ Aids in material selections/conditions

Con's -

- ☹ Long processing times
- ☹ Experience analysts needed for simulations
- ☹ Correlation for wrinkling unknown
- ☹ 3-4 month flow (at best) for baseline results
- ☹ Too slow for parametric feedback